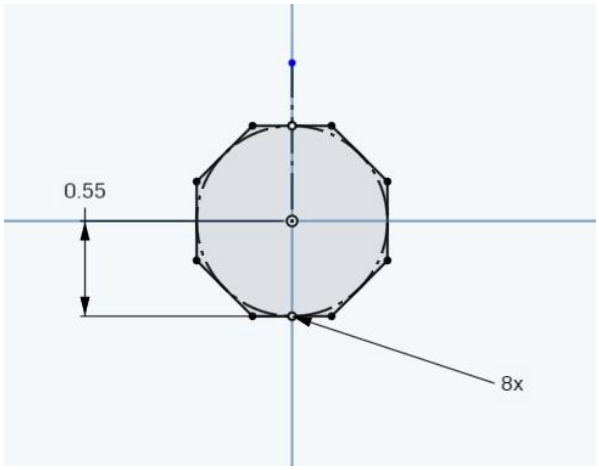
In this project students will be tasked with designing the space needle in onshape, this project is designed to be a intermediate level project that will get students familiar with basic tools in onshape such as using the sketch feature and creating parts in the parts studio and putting it all together into a final assembly

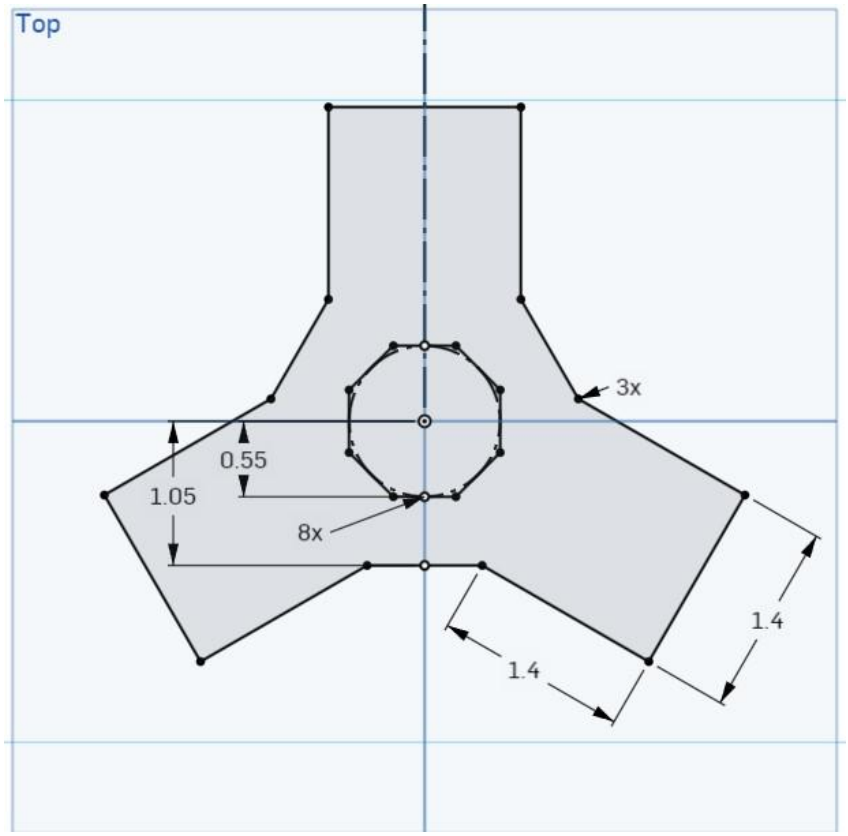
The Space Needle

Part 1: Part Studio 1 and Assembly 1

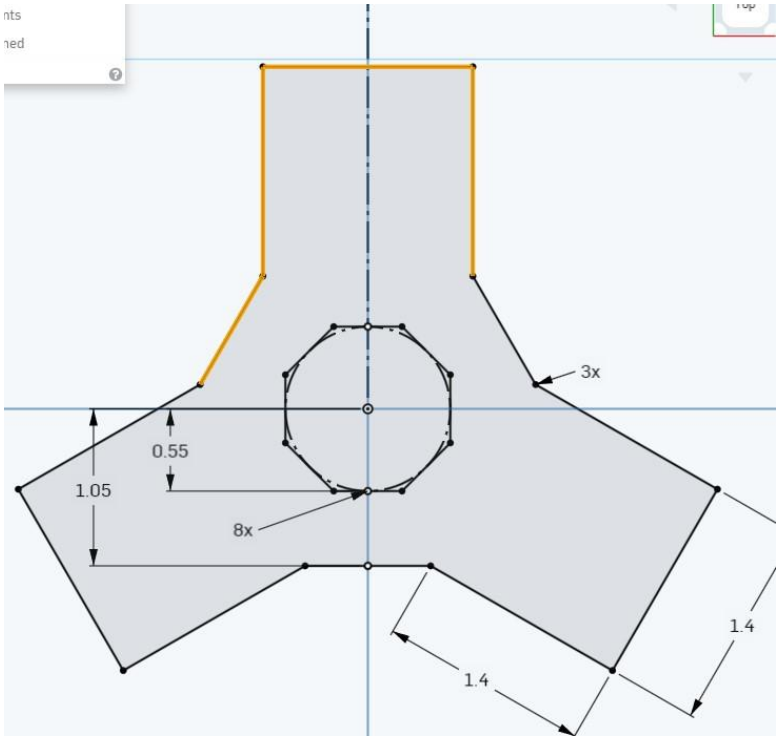
1. Open Onshape and log into your account, go to your dashboard and click create in the top left corner then click document, name your document: SpaceNeedleProject
2. In the top left corner click create new sketch, click the “top” plane to draw your sketch on.
3. Click the “inscribed polygon” tool on the tool bar and create an octagon with an area of around 1in (this doesn’t have to be exact just around) create this octagon centered on the origin.
4. Next we are going to want to fully define this shape, to do this we are going to use the dimension tool and take the radius of of our octagon
5. Then we are going to use the construction tool and draw a line from the origin up a little bit past the edge of our octagon
6. Next create a midpoint on the top line of your octagon, onshape will automatically choose the center of the line when using the point tool.
7. Now we are going to choose our construction line and our midpoint and use the coincident tool to attach them together, this ensures the octagon now has 0 degrees of freedom.
8. You will know your sketch is fully defined because it will turn black.
9. Here is what it should look like:



10. Shift-e or click on the “extrude” button on the toolbar, drag the octagon up until you get to 12 in (or alternatively type in 12 where it asks for Depth in the extrude 1 panel) then click the green check mark when finished
11. Name this new part, stem.
12. Create a new sketch- click top plane to draw the sketch on, here we want you to recreate this shape. With the dimensions shown in the picture

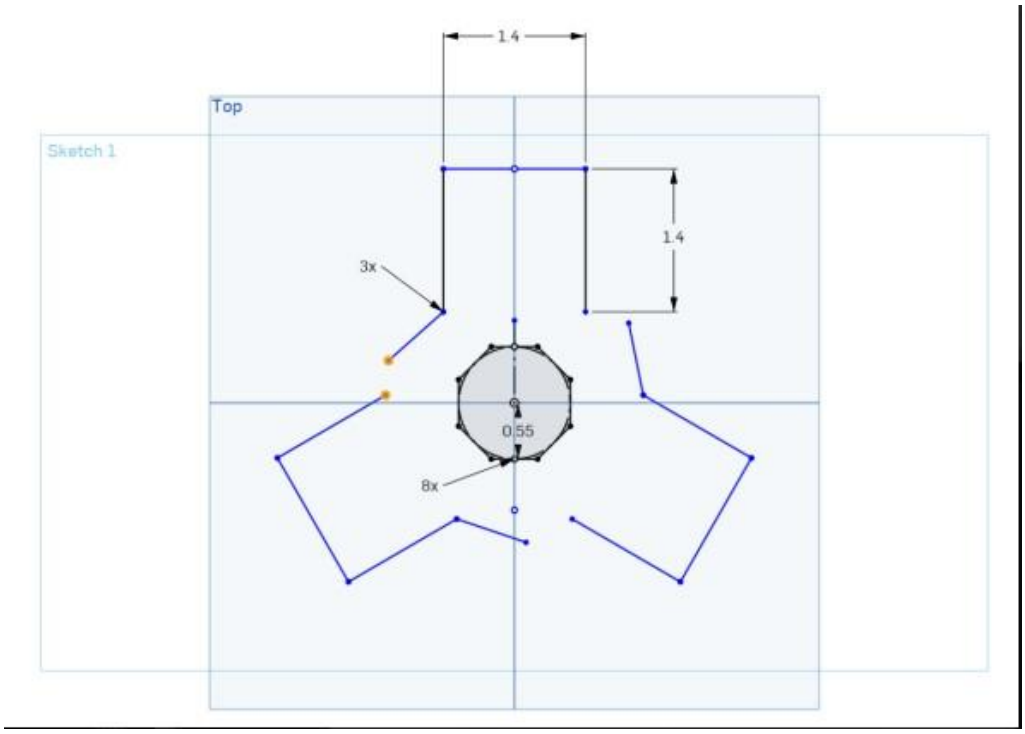


13. To start, take the Octagon from your first sketch and insert it into the new sketch, to do this you are going to want to use the project/convert tool in your toolbar to take our previous sketch and paste it into sketch 2.
    a. make sure this is centered on the origin
14. Now you are going to draw a straight line using the line tool across the center axis above the octagon, use the dimension tool to check how long your line is and make sure it is 1.4 in long.
15. Now the two sides coming down from this top line are also 1.4 in long, (this is basically a square but without the bottom line)
16. Now create a slanted leg coming off the left line, it might seem suspicious, but the length and angle of this line doesn't matter too much for right now and you'll see why in a second. For now try and make your three parts looking like the highlighted ones in the picture below

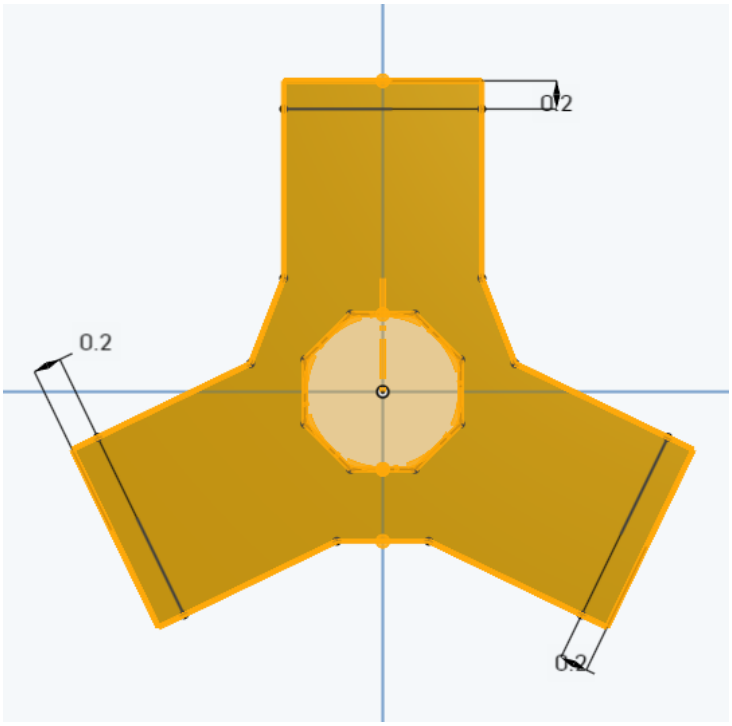


17. Now we are going to use the Circular Pattern tool. To get to this tool click on the drop down arrow next to the fillet tool and click circular pattern.
18. Click on your 4 sketch entities from your sketch and use a circular pattern which will -- as one could guess -- create a circular pattern around the origin. Now you will have the 3 legs of the platform but they aren't connected

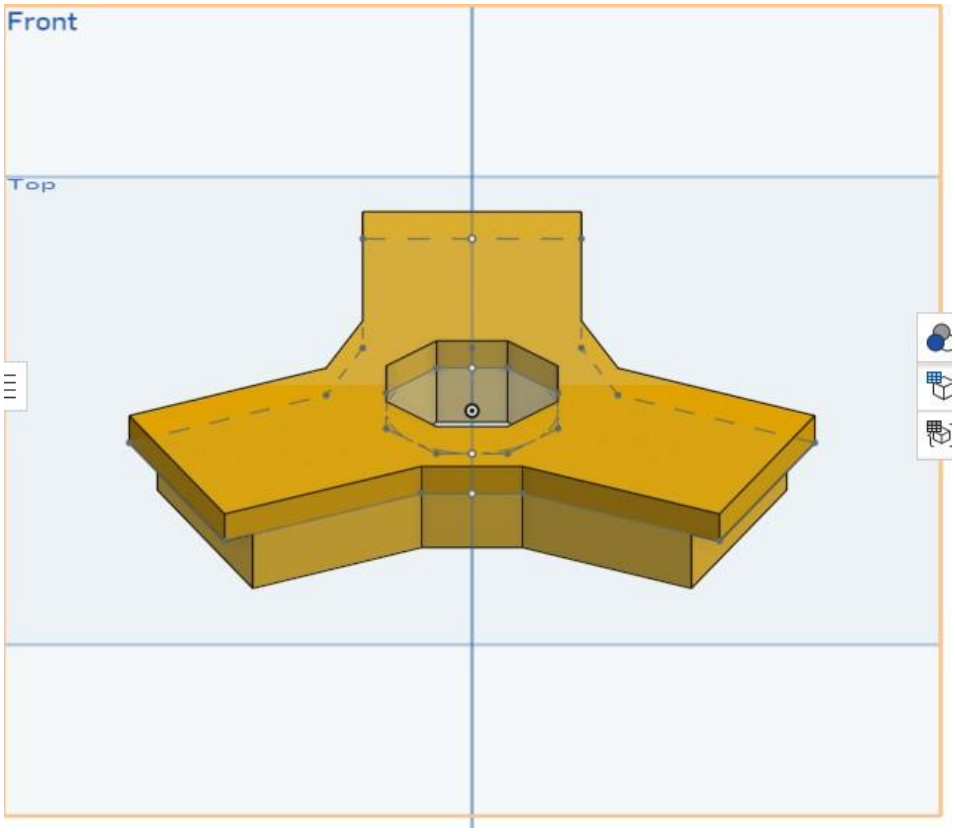
19. To connect these legs click on the two points that are highlighted below and use the coincident tool to stick them together, this should connect all of the edges together automatically since we created a circular pattern



20. Make sure your sketch is fully defined before moving on.
    a. To do this you are going to take the distance from the origin to the bottom horizontal line. Your sketch should be fully defined by now
21. Next we are going to extrude this shape to a depth of 0.20 in
22. Then we are going to create a new sketch that will be drawn on the face of sketch 2
23. You want to create a shape that is the exact same as shape 2 however the three side lengths as pictured need to be 0.2 inches in from the shape before here is a picture:

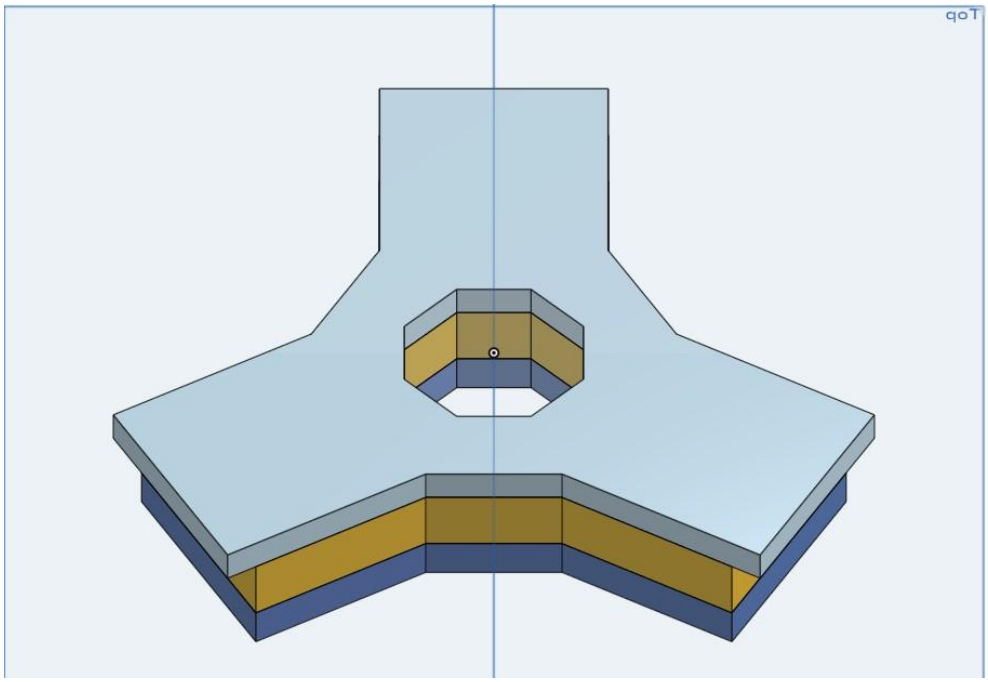


24. Use project geometry to project one of the 3 angled lines from sketch 2 into your new sketch 3
25. Now create the same square as before except instead of 1.4 inches it is now 1.2inches, use the same technique as in the last part (circular pattern and coincidence) to create your shape
26. Fully define your shape by dimensioning the distance from the origin to the bottom horizontal line.
27. Now we want to extrude this shape down 0.4 inches
28. Your part 2 should look like this

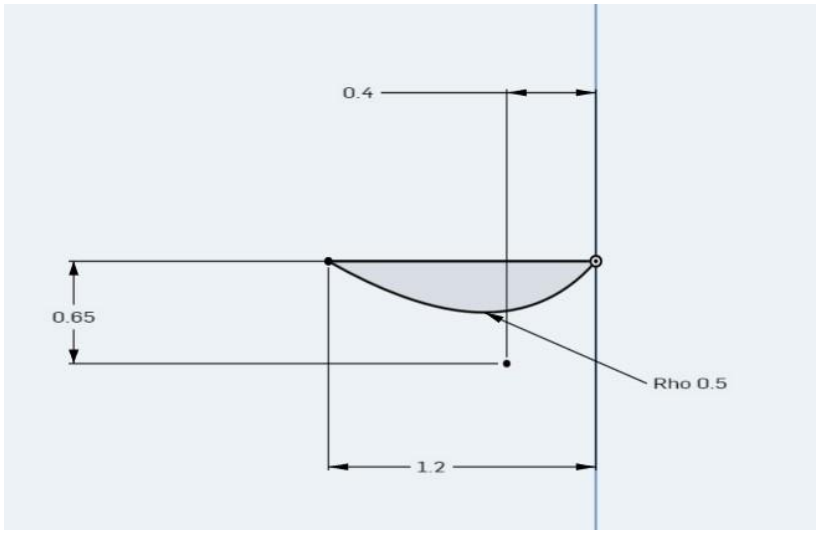


29. We are now going to want to create another sketch that is the exact same as the sketch we just created, to do this we are going to draw this sketch on the face of the previous extrude (extrude 3)
    a. To copy and paste the exact sketch use the project/convert tool in your tool bar to take our previous sketch (sketch 3) and put it in sketch 4.
30. Now we want to extrude this by 0.2 in
31. Here we are going to change the color of our platforms to make it more clear what part differs from the other.
32. Go to the right hand side of your screen and click the three circles, here you should be able to click on the different parts of your part studio and change the color associated with them.
    a. Make the top thin part of the platform light blue
    b. The longer smaller platform extruded from this yellow
    c. And the final bottom part of the platform dark blue.

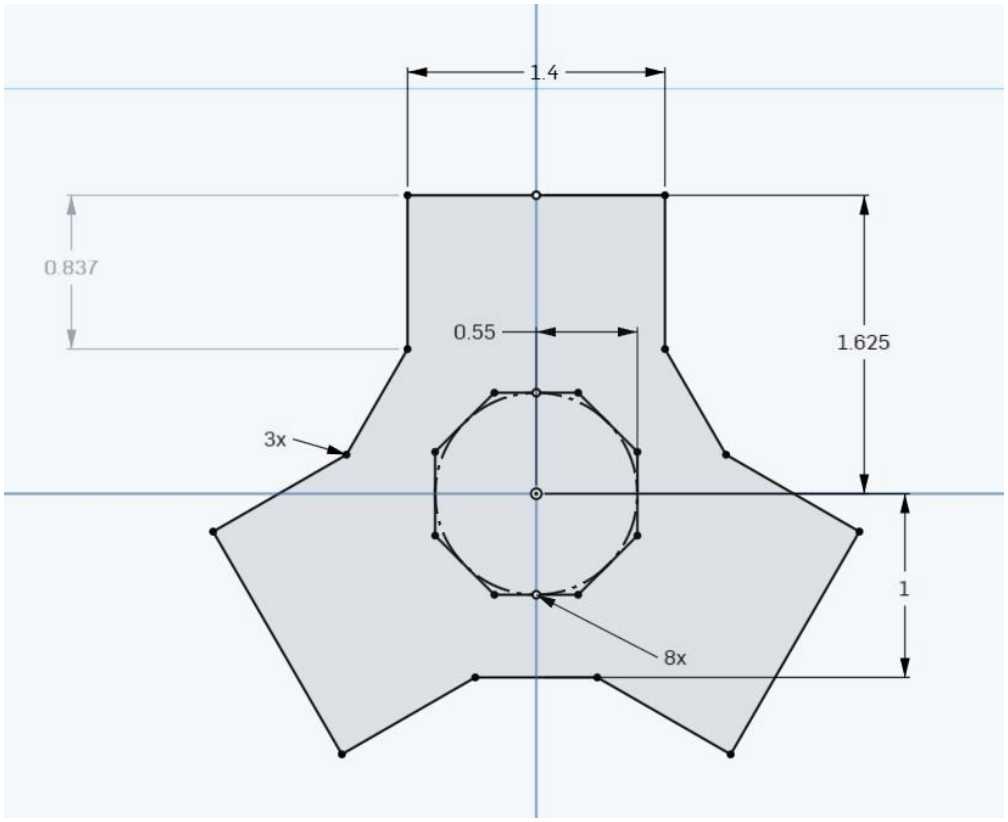
33. Your part studio should now look like this:



34. Next we are going to want to create arches to put on the bottom of this platform, to do this create a new sketch and use the conic tool with the dimensions below with the rightmost edge on the origin



35. Next you'll want to extrude this arch to a depth of 0.2 in and rename this new part arch
36. Now we are going to make the middle platform, this platform is the same as the bottom platform except with these dimensions:

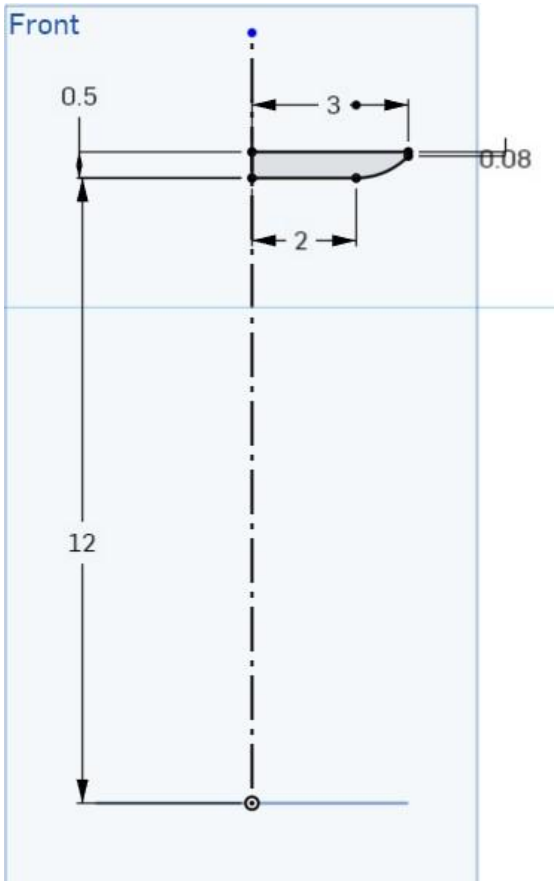


37. Use the same process as we did in the other two sketches (you should be a pro at this by now!)
38. Make sure your sketch is fully defined and rename it middle platform.
39. Now you should have 6 parts in your part studio one.
40. Name this part studio Lower platform and stem
41. Now we are going to create a new part studio

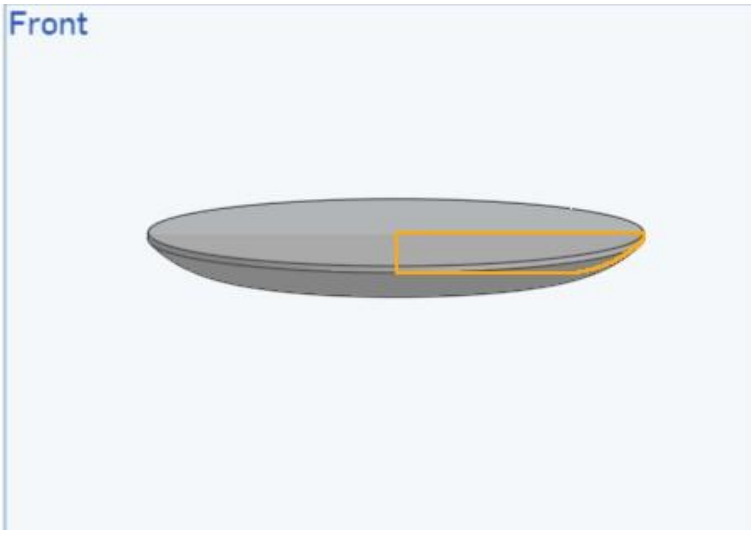
Part 2: Part Studio 2

In this part of the assignment we are going to be creating the “legs” of the space needle along with the base of the top

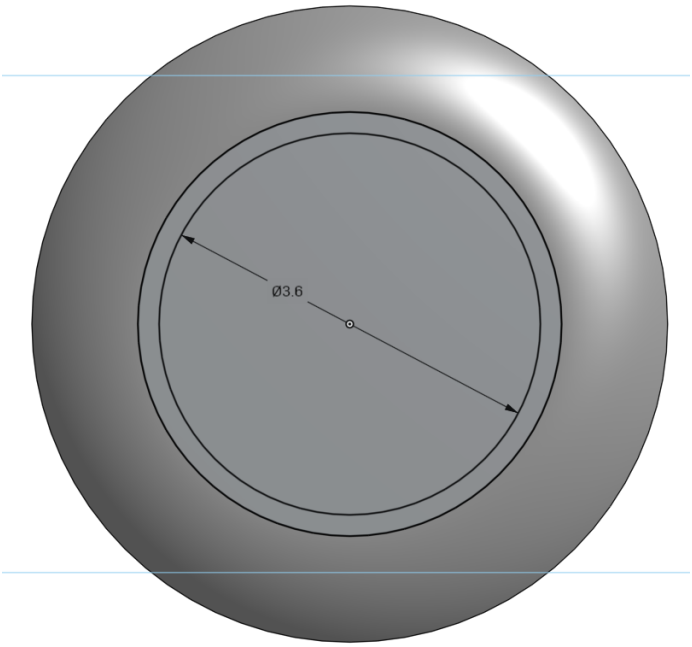
1. Create a new part studio and begin a new sketch on the front plane.
2. The first thing you are going to want to do is create a vertical construction line along the origin, make this line super long (+15in)
3. Now you are going to want to recreate the shape below with the dimensions below
    a. To make this sketch fully defined you’re going to have to use the tangent tool
    b. Try your best to do this on your own! You got this!



4. Once you finish your sketch you’re going to want to use the revolve tool.
5. The revolve tool asks you to define the dances and sketch region to revolve, here you should select the face of sketch 1.
6. The revolve tool now asks you to define the axis on which you are going to revolve around, here you are going to want to select the construction line you created in sketch one. Your part should now look like this.

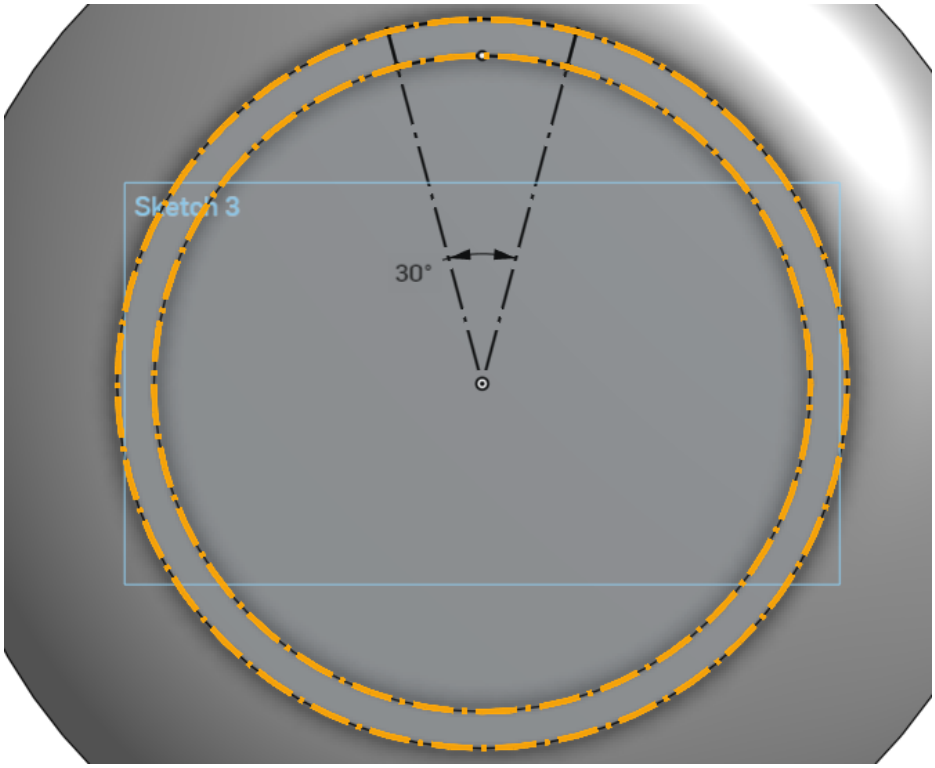
7. 

8. Now we are going to create a new sketch, this one is going to be on the bottom face of revolve one.
9. You are going to want to create the shape below



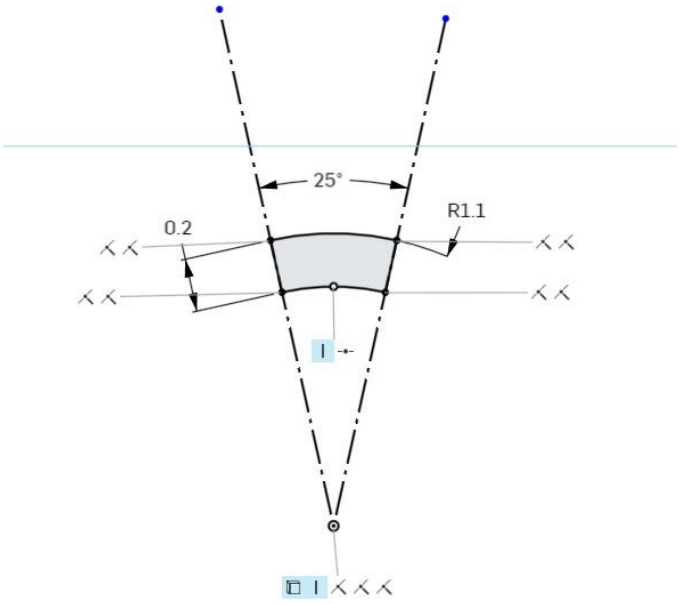
10. To do that you will use the project geometry tool to copy the big circle (face of the revolve) from sketch 1
11. Then use the circle tool to create a circle of diameter of 3.6in centered at the origin
12. Make sure your sketch is fully defined
13. Now you are going to extrude this shape by 0.2 in
14. Now we are going to create another sketch, this one is going to be on the face of the extrude we just did

15. Project the outer rim just like we did in the previous sketch and now also project the inner rim we just created. Make both of these construction lines
16. Use the centerpoint arc tool to create an arch on the outer circle: to do this click the origin to make that the center of your circle, then click two points on the outer circle -- roughly where they are shown in the photo below but this doesn't have to be exact
17. Create construction lines from the endpoints of that arch to the origin
18. Now you are going to want to draw a line from the endpoint of the arc along the construction line to the *inner circle*
    a. Do this for both endpoints

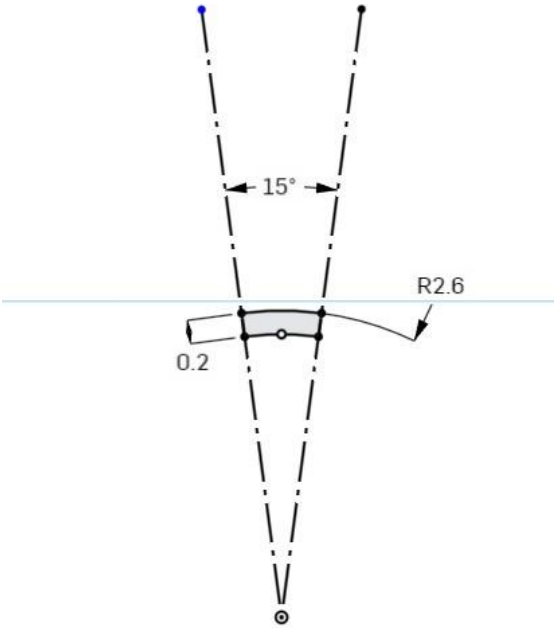


19. Use the center point circle tool and connect the endpoints on the inner circle to create an arc also centered on the origin
20. Now use the point tool to place a midpoint along one of the arcs and use the vertical constraint to make this vertical with the origin
21. Use the dimension tool to make the angle between the two construction lines making the arc 30 degrees
22. Make sure your sketch is fully defined and we are finished with the sketch! *high five*
23. We are going to create a new sketch however we need to create a new plane to draw this sketch on first.
24. To do this click on the "plane" tool on the toolbar above
25. Select the top plane as the entity and offset it 8 inches above the top plane

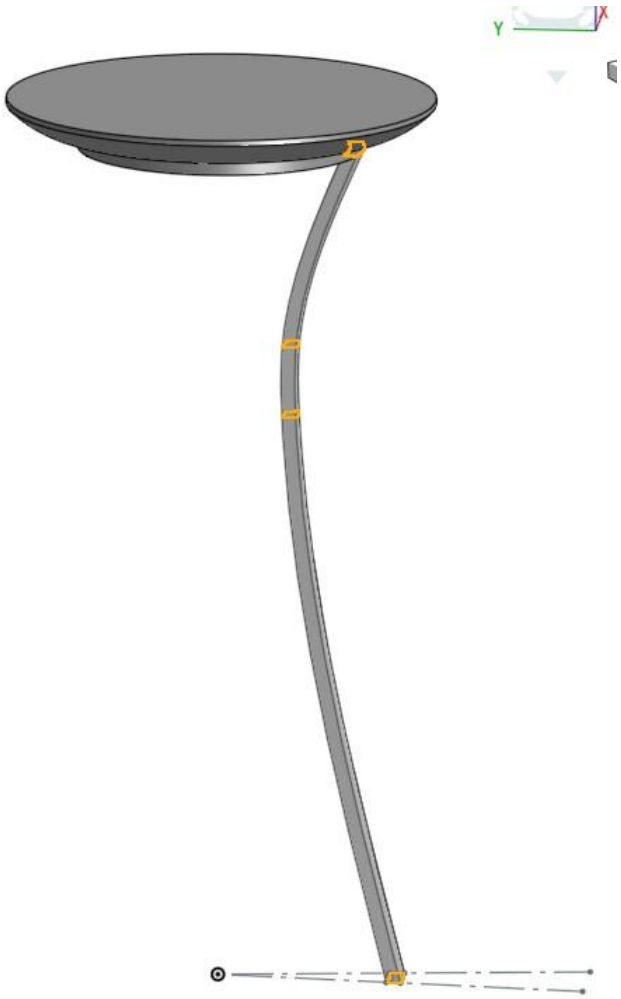
26. Now that we have this super awesome new plane, we are going to create a new sketch on this plane.
27. You are going to want to use the center point arc to create this shape,
28. Like you did in the previous sketch make the V portion of the arc construction lines
29. You should be familiar with this shape since we just made it in the previous part, here is what it should look like.
    a. I included the constraints I required to make my sketch fully defined



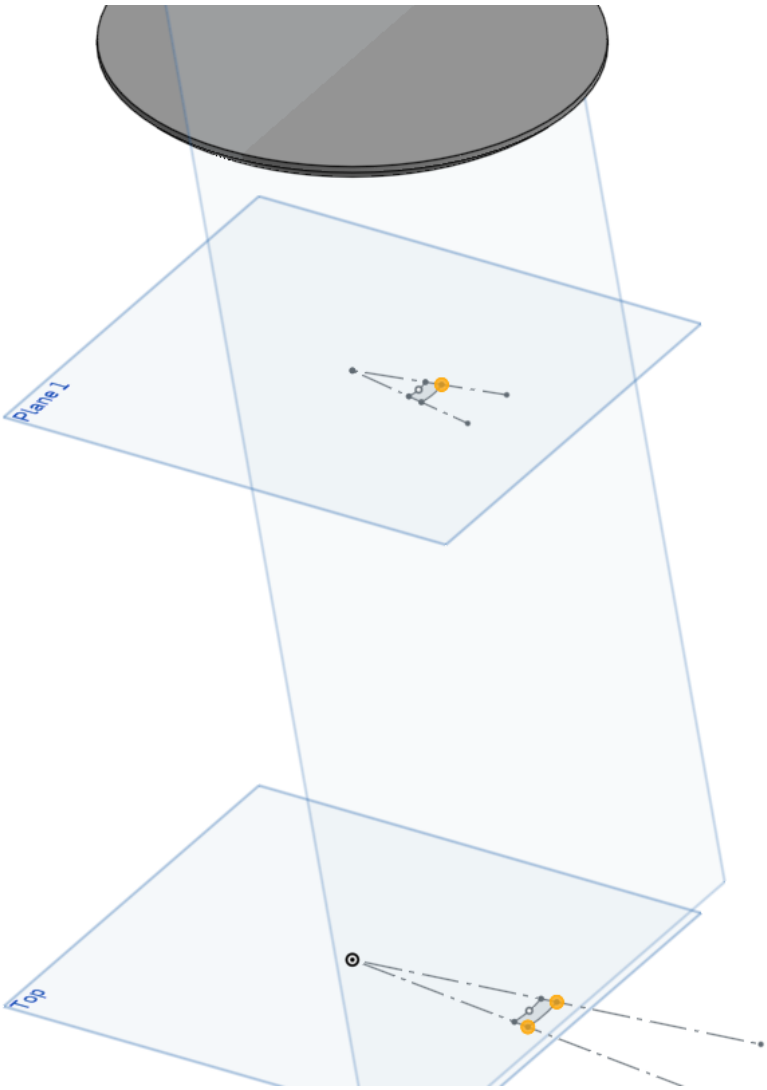
    b.
30. *****Make sure that this sketch is lined up with the arc you made in sketch 3*****
31. Now we are going to create another plane, this plane is going to be offset 1in above the plane you just created.
32. Now we are going to create a new sketch on this plane
33. You are going to want to create an exact copy of the sketch we made in step 29, do this by using project/convert tool
34. Create a new sketch on the top plane (you should be on your 6th sketch by now)
35. Here you are going to use the same techniques we have done in the previous 3 sketches to create another arc with the dimensions below
    a. Your sketch fully defined should look like this:



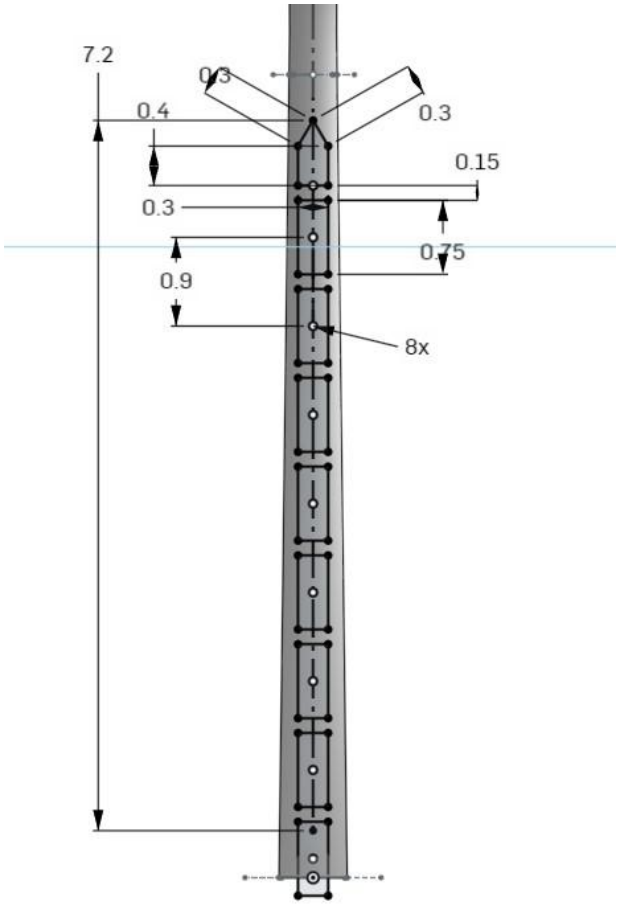
36. Now… we get to use the loft tool !!!!!!!!! :D
37. Click the loft tool, when you use this tool you're going to have to define the profiles on which the loft will be defined. Here you want to select the 4 arc sketches we just made from the top down,
    a. So the first sketch you're going to select will be the face of sketch 3, then the face of the arc created on the sketch below that one etc…
    b. By doing this we are effectively creating the "legs" of the space needle, we needed to create this many sketch to be able to control the curve of the legs
38. Once you select the 4 sketches confirm the loft and your part studio should now look like this:



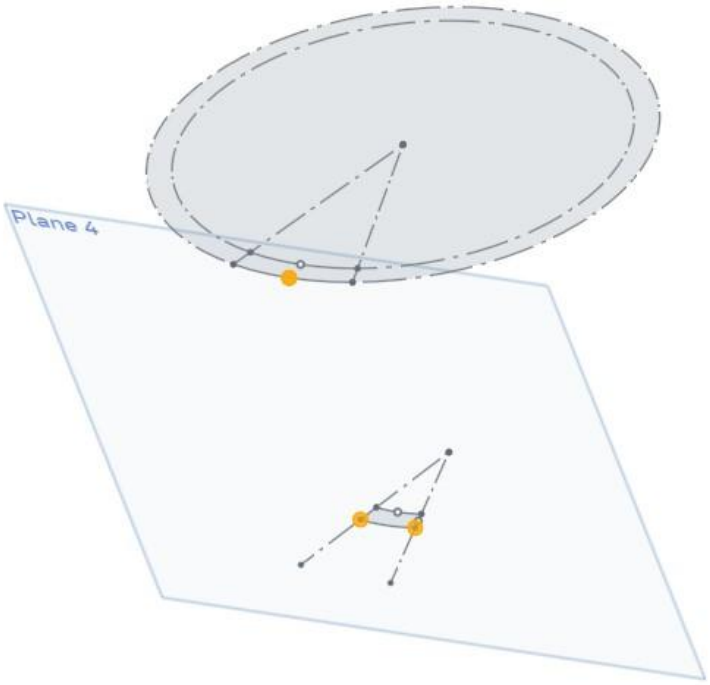
39. Nice! It's really coming together now isn't it (:
40. Create a new sketch on the front plane
41. Create a very long construction line (+15in, make sure it clears the front plane table)
    a. This is going to be your axis of rotation, so name the sketch appropriately
42. Create a new plane, this one is a bit more complicated. We will use the Three Point method to define the plane
43. You're going to want to create this plane using an endpoint from the outer arc on sketch 4, (this is the second arc you created, on the first new plane you created - plane 1-)
44. Now use the two endpoints on the outer arc on the 4th sketch you made(the most recent sketch -- the arc at the bottom)
45. Refer to the picture below to see what your plane should look like,
    a. We are creating the holes on the legs on this plane so we need the plane on which we draw our sketch to be parallel to the face of the leg



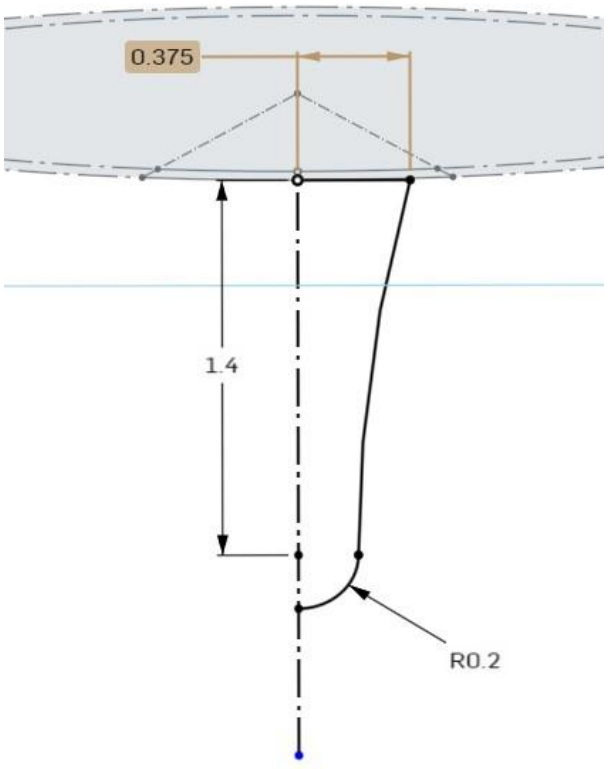
46. To create the holes on the legs you are going to want to recreate the sketch below with the given dimensions.
47. Make the top triangle hole then the rectangle below it, then use the linear pattern tool
    a. Click the center point of one rectangle and the center point of another one and use the vertical constraint to make sure these line up
    b. Use the equal constraint with the bottom of the triangle and the top of the rectangle to make sure the distance is the same.
    c. Make the number of instances 8 in the linear pattern



48. Once your sketch is fully defined, extrude your sketch by 2 in, and click symmetric instead of blind, in addition, this time you're going to remove instead of add when using the the extrude tool since we want these to cut through the legs
49. Great job, now we are going to create another new plane
50. Use the drop down in the plane menu and select 3 point
51. Use the midpoint of the outer arch of sketch 3 (the very top) and the two end points of the arch on sketch 5 (the second from the top)
52. Your plane should look like this

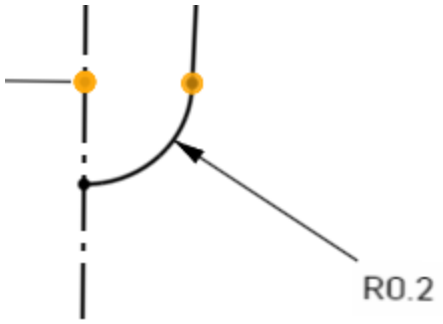


53. Create a new sketch on this plane and create a construction line down the center that's longer than 3in at least
54. Project the midpoint of the big arch from sketch , then create a horizontal line starting at that point with the dimensions shown below.
55. Using the 3 point arc tool, create an arc starting from the outside of the line you just drew and draw it down and make sure it is curved like the picture, we will adjust the length later



56. Now use the tangent arc tool and create a 90 degree arc with the midpoint on the construction line

a. To make it 90 degrees make the point, constrain the highlighted points below to be horizontal



57. Now use the mirror tool, select the construction line as the axis and mirror everything across this line.

58. Fully define the sketch

59. Now you are going to want to extrude this sketch, this time using the “up to face” parameter instead of blind.

a. Select the face of extrude 1 for “up to face” again you will be using remove instead of add

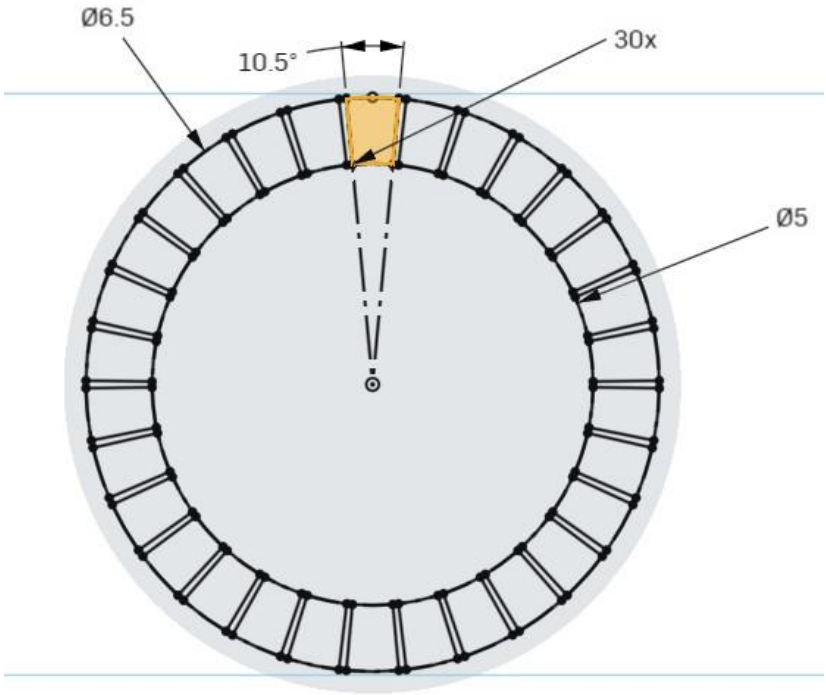
60. This is what your part studio should look like now

61. Now you might be like wait! That's only one leg!!!!! But don’t worry, we have another tool that can help us!
62. We are going to use the circular pattern tool! Remember that axis of rotation we made a while ago? We are going to use that to be our axis of rotation for this circular pattern.
63. Select part 1 as your entity to pattern, then click for your axis of pattern the edge of the axis of rotation you created.
64. Set the angle to 360 degrees, and the instance count to 3, you should now have 3 legs and the top base for your space needle!
65. Phew, take a breather (:

Part 3: Part Studio 3 and 4

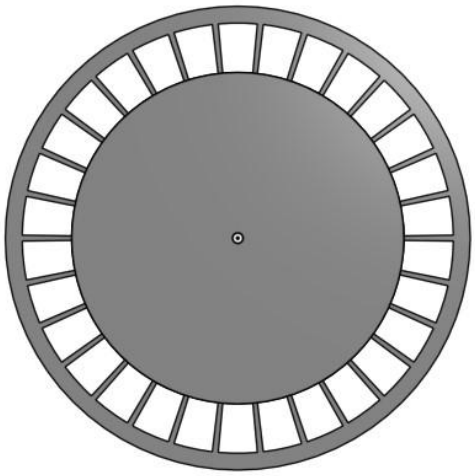
1. Welcome
2. Create a new sketch on the top plane
3. Create a circle of diameter 5 centered about the origin, fully define this sketch
4. Extrude this sketch to a depth of 0.5 in
5. Create a new sketch, again on the top plane

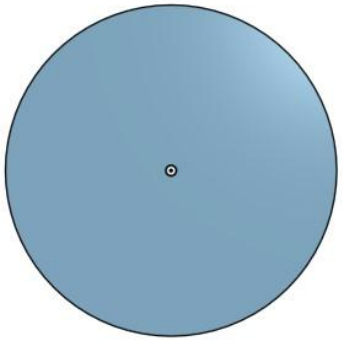
6. Create a circle with a diameter of 7 in
7. Fully define the sketch
8. Extrude this sketch by 0.3 in
9. Create a new sketch, this one is going to be on the face of your most recent extrude (extrude 2)
10. Now you are going to create 2 construction circles one with a diameter of 6.5 and the other with a diameter of 5, centered about the origin (refer to picture below)



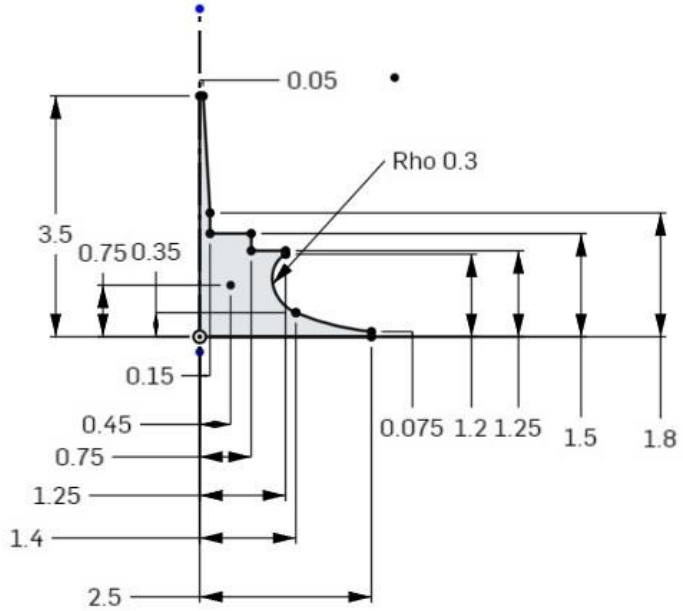
11. Use the center point arc tool and create an arc on the outer circle
12. Make construction lines to the origin from the edge points of this arc
13. Then create actual lines from the outside circle to the inside circle along these construction lines
14. Then create an arc on the smaller circle connecting the points of those lines.
15. Now put a point on the midpoint of the outer arc and vertically constrain it to the origin
16. Make the angle between the construction lines 10.5 degrees as pictured above
17. Now you will use the circular pattern tool the box highlighted above and select 30 instances equally spaced
18. Your final sketch look look like the one above, fully define your sketch before moving on

19. Now we are going to extrude this sketch, you are going to use the remove tool of extrude and choose up to next.
    a. You can also use blind and set it to be a certain amount of inches, just as long as the holes go through the entire disk
20. You are now going to create another sketch, this one is going to be on the face of extrude.
21. Create a circle centered at the origin of radius 5
22. Extrude this by 0.5 inches
23. Create a new sketch, this one is going to be on the top plane
24. Create a circle centered at the origin of radius 5
25. Extrude this by 0.5 in
26. (you should have two identical parts)
27. Name one of these identical parts blue circle and the other grey circle and change the colors accordingly

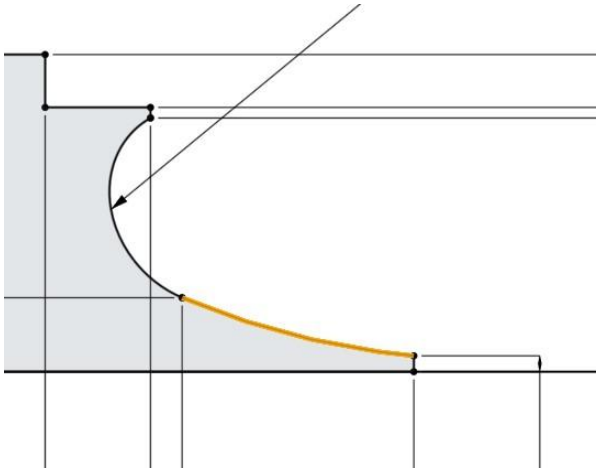




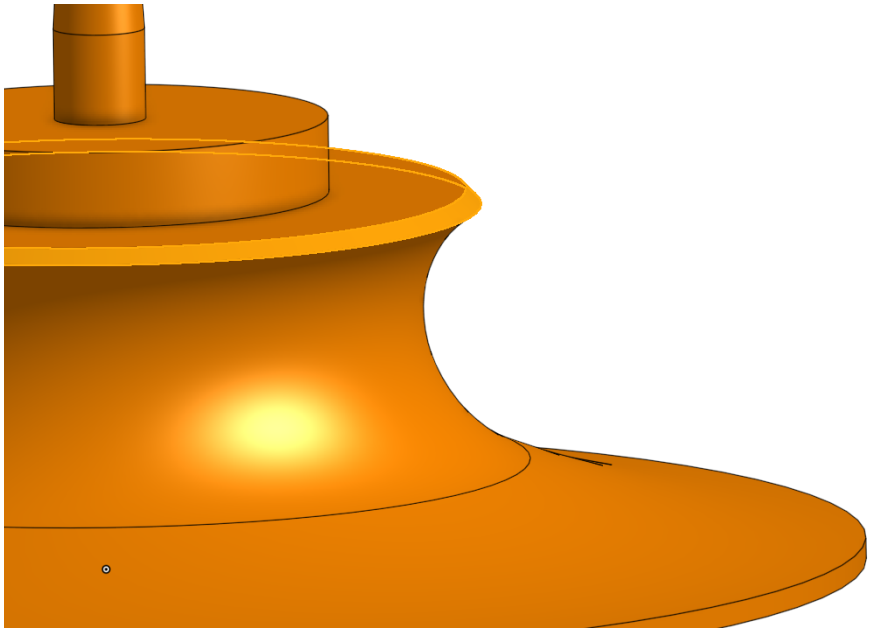
28. You should have 3 parts in your part studio, one being a grey circle, one being a blue circle, and the final one being a larger grey circle with holes cut through it, use these photos for reference
29. We are done with this part studio, rename it middle top or something you are able to refer back to and recognize
30. Create our final part studio
31. Here you are going to create a sketch that looks like the one below with the dimensions listed



32. To do this you are basically going to use the line tool and the conic tool as well as one 3-point arc -- this is in between the conic and the bottom, the 3-point arc is highlighted in the picture below.



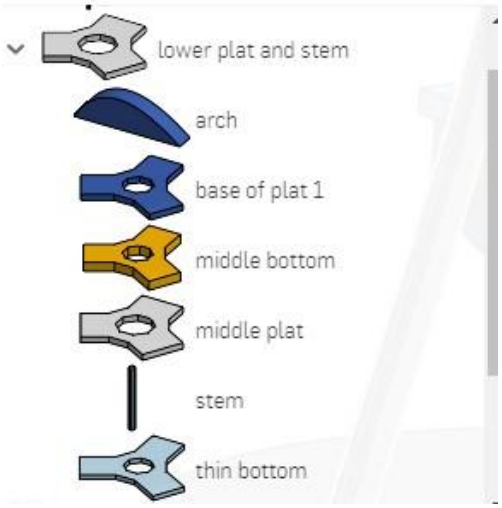
33. Once you finish make sure your sketch is fully defined and then use the revolve tool
34. Here you will select the faces of sketch 1 to be your region to revolve and the revolve axis to be the edge of sketch one (the line down the center)
35. Set the revolve to full
36. Now we are going to use the chamfer tool
37. Select the edge of revolve one (the highlighted area below) , then set it to equal distance and the distance to 0.05 in



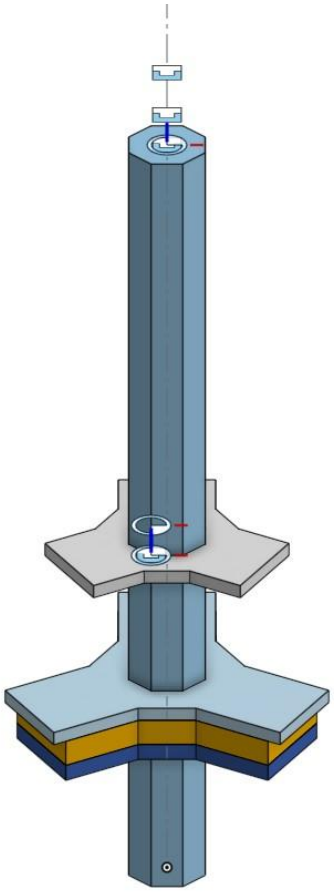
38. We are now going to create our final part which will be a circular base for the space needle to sit on
39. Create a new sketch on the top plane, using center point circle create a circle about the origin with a diameter of 8in
40. Extrude this by 0.25 in
41. Name this part base
42. We are now done with this part studio, rename it “top and base” or something you are able to recognize it with

Part 4: Assembly 1

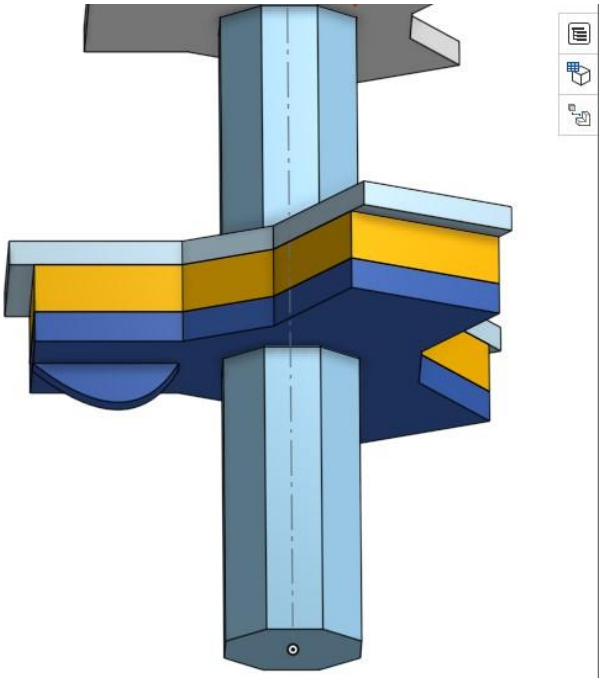
1. Here we are going to assemble all of the different parts we have made in assembly 1
2. Here we are going to click insert at the top corner of our assembly
3. Click on the drop down arrow next to your first assembly and select the stem
4. Click on the green checkmark at the top of the screen to automatically insert your part at the origin
5. Next we are going to add in the “thin bottom”
6. - refer to the picture as you might have named your parts something different

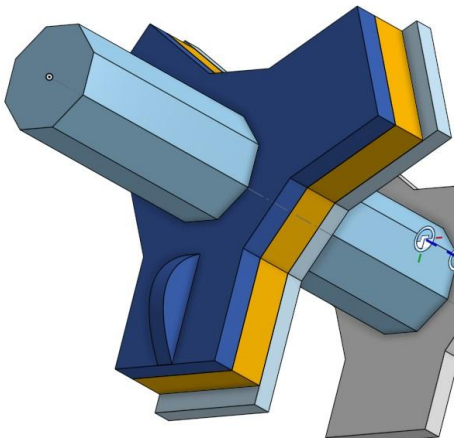


7. Click on the green check mark
8. Now we are going use a mater connecter to attach these two parts together
9. Go to the top of your screen and click on fastened mate
    a. This mate removes all degrees of freedom between two parts which is exactly what we want since we don't want the base of our space needle to be able to move up and down on the stem, later on in this class you will learn about the different mate connectors, but for this project we want everything to be unable to move.
10. Now you are going to click the center mate connector of your platform and connect it with the center mate connector of the base of the stem, adjust the height with the offset tool in the fastened drop down menu, you're going to adjust the z axis by 3.25 in
11. Repeat these steps with the other two platforms we built in that part studio, adjusting the z axis and then do it with the middle platform, adjusting the z axis until it is a little below halfway up the stem
12. This is what your assembly should look like right now

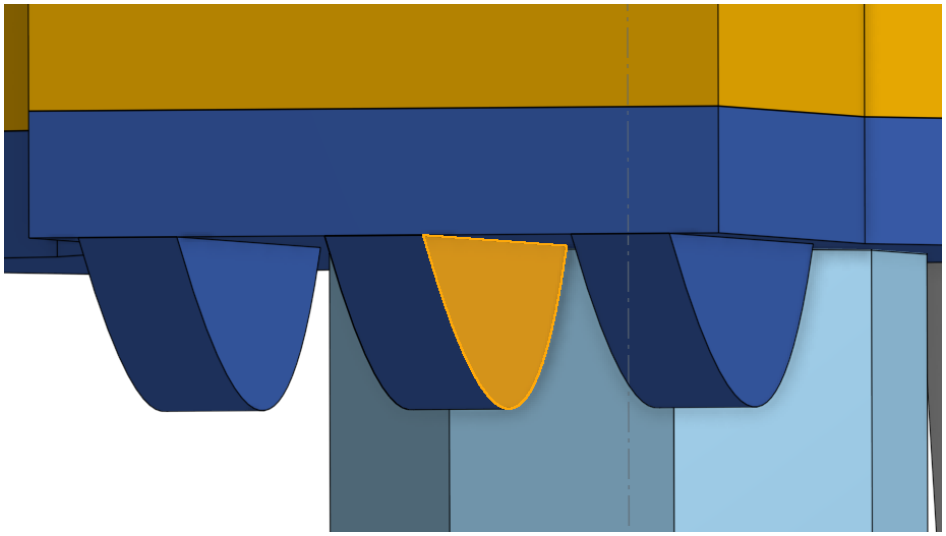


13. Now we are going to insert the arches, to start this go to insert parts and assemblies and click on the rectangle and square icon that says sketches,
14. Here you are going to go to your second part studio and choose your axis of rotation sketch, click the green checkmark at the top and it will automatically center this at the origin
15. Now you are going to go and insert the arch you made from your first part studio
16. You are going to want to fasten the midpoint of the edge of the arch to the midpoint of the edge of the bottom of the lower platform.
    a. This sounds a bit confusing but refer to the photos below to see how it should look in the end

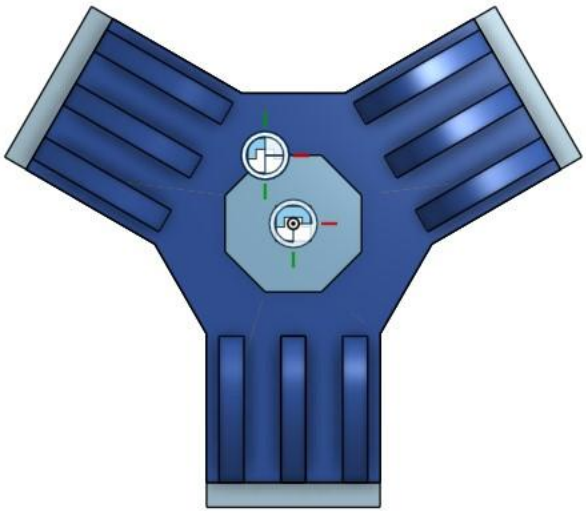




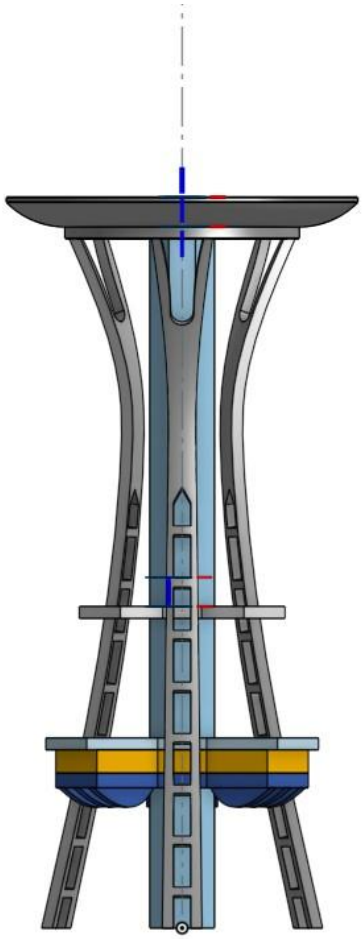
17. Now you are going to use the linear pattern tool and select the arch as your instance
    a. Then for the direction choose one of the flat faces of the arch
    b. Choose distance as 0.5 inches and create 2 instances
        i. This should give you 2 equally spaced arches
    c. Now select second direction and choose the other flat face of the arch (refer to the picture below to know exactly which faces of the arch to select
    d. Make this distance 0.5 in and the instance count 2
        i. Double check the direction to ensure that the arch is on the platform
    e. You should have 3 arches on one of the edges of the platform now



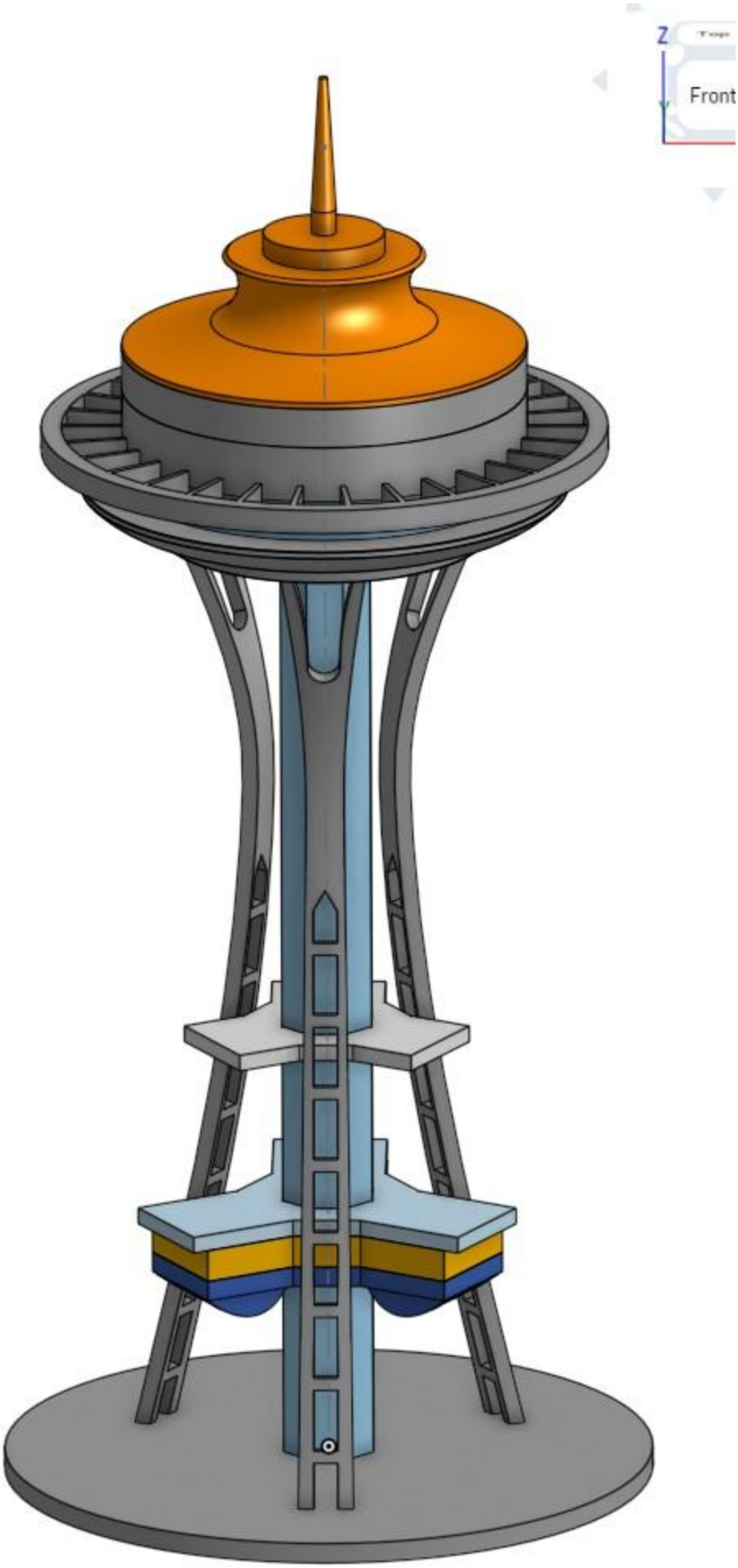
18. Now select the circular pattern tool and the three arches we just created as the instances
19. Make the axis of pattern the axis of rotation sketch we inserted
20. Make the instance count 3 with equal spacing
This is what you should have on the bottom of your platform if done correctly



21. Now we are going to insert the legs and top base from our part studio 2
22. To do this, click on insert then click on your part studio 2 and then the green checkmark, this should insert the entire legs and top base into the assembly.
    a. It should be noted here you should have 3 parts inserted one for each of the legs, however you will see that the top base is connected to each of the legs so you might be worried about inserting 3 of the top bases, that is okay since we are going to fasten them all to each other
23. Now to fasten these to the stem click on the fastened mate, click the center mate connector of the stem and the center of the top base
    - Repeat this process for all 3 parts in this part studio, your assembly should now look like this:



24. Now we are going to go to our 3rd part studio and insert the 3 parts we created here,
25. You are once again going to use the fasten mate, where you will fasten the center of the blue circle to the center of the top base mate connector
26. Then repeat this process for the next 2 parts in this part studio, using the fasten mate on the center mate connector of the part you are inserting and the center mate connector of the part below it.
27. Go to your 4th part studip containing the top and the base of the space needle,
28. As you might have guessed you are going to insert the top and use the fsten mate to connect the center mate of this part with the center mate of the part below it
29. Then lastly for the base you will also you a fasten mate and connect the center mate of the base to the center mate of the stem at the bottom of your assembly
30. THEN TADA! You’re finished!!! This is what your final space needle should look like in all of its glory.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